



Aalborg Universitet

AALBORG UNIVERSITY
DENMARK

Computational Fluid Dynamics in Ventilation Design

Nielsen, Peter V.

Published in:

Clima Systems Towards a new Equilibrium Between Mankind and Environment

Publication date:
2008

Document Version
Publisher's PDF, also known as Version of record

[Link to publication from Aalborg University](#)

Citation for published version (APA):

Nielsen, P. V. (2008). Computational Fluid Dynamics in Ventilation Design. In C. M. Joppolo (Ed.), *Clima Systems Towards a new Equilibrium Between Mankind and Environment* (pp. 617-626). Aicarr.

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- ? Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- ? You may not further distribute the material or use it for any profit-making activity or commercial gain
- ? You may freely distribute the URL identifying the publication in the public portal ?

Take down policy

If you believe that this document breaches copyright please contact us at vbn@aub.aau.dk providing details, and we will remove access to the work immediately and investigate your claim.

Computational Fluid Dynamics in Ventilation Design

La dinamica modellistica dei fluidi nel progetto di ventilazione

PETER V. NIELSEN

Aalborg University, Aalborg, Denmark

ABSTRACT

This paper is based on the new REHVA Guidebook *Computational Fluid Dynamics in Ventilation Design* (Nielsen *et al.* 2007) written by Peter V. Nielsen, Francis Allard, Hazim B. Awbi, Lars Davidson and Alois Schälén.

The guidebook is made for people who need to use and discuss results based on CFD predictions, and it gives insight into the subject for those who are not used to work with CFD. The guidebook is also written for people working with CFD who have to be more aware of how this numerical method is applied in the area of ventilation. The guidebook has, for example, chapters that are very important for CFD quality control in general, and for the quality control of ventilation related problems in particular.

A large number of CFD predictions are made nowadays, and it is often difficult to judge the quality level of these predictions. The guidebook introduces rules for good quality prediction work, and it is the purpose of the guidebook to improve the technical level of CFD work in ventilation.

RIASSUNTO

Questa memoria si basa sui contenuti della nuova REHVA Guidebook *Computational Fluid Dynamics in Ventilation Design* (Nielsen *et al.* 2007) scritta da Peter V. Nielsen, Francis Allard, Hazim B. Awbi, Lars Davidson and Alois Schälén.

La guida è realizzata per quelle persone che devono utilizzare e discutere risultati ottenuti mediante CFD ed allo stesso tempo fornisce un supporto per coloro che non sanno utilizzare tale metodologia.

La guida è anche pensata per le persone che utilizzano il CFD e che desiderano conoscere più approfonditamente questo metodo numerico applicato al settore della ventilazione. Per esempio essa contiene capitoli che sono molto importanti per il controllo della qualità del CFD in generale ed in particolare per il controllo della qualità

dei problemi connessi alla ventilazione.

Al giorno d'oggi sono realizzate molteplici simulazioni mediante l'utilizzo di CFD ed è difficile poterne giudicare il livello di qualità. Questa guida, dunque, introduce le regole per realizzare simulazioni corrette al fine di migliorare il livello di preparazione tecnica di chi utilizza CFD nel settore della ventilazione.

1. INTRODUCTION

The chapters *Mathematical background and Turbulence models* in the REHVA Guidebook give a short introduction to the theory behind the methods used in CFD modelling. The fundamental transport equations are discussed with emphasis on ventilation applications, and descriptions are given for two-dimensional geometry to simplify the concepts. A user of CFD predictions must have some knowledge of fluid mechanics. It is important to understand conditions such as: laminar flow, turbulent flow, steady flow, time dependent flow, etc., both in connection with CFD and also with measurements in rooms for validating the predictions. The two chapters give some insight into all of these conditions.

The turbulence model is specially an important aspect of CFD. It is obvious that room air flow will be turbulent because of geometry and practical velocity levels, but it will not always be a fully developed turbulent flow. Some of the widely used models are discussed such as the k - ϵ model, the SST model, and the Reynolds Stress model. The Large Eddy Simulation is also shortly mentioned.

The chapter on *Numerical method* illustrates the structure of a CFD program, and it demonstrates much of the experience a user should have in using a commercial program. Most of this chapter is based on a one-dimensional theory. The use of a one-dimensional analysis made it possible to understand, by hand calculation, many concepts and issues such as: order of accuracy, necessary number of grid points, wiggles in the solution, iterations, divergence, etc. This is demonstrated with a convection-diffusion equation which is solved with the use of different schemes at different velocity levels.

The chapter on *Boundary conditions* is especially important in the ventilation area. Often the flow in a room is determined by small details in the diffuser design. This means that a numerical prediction method should be able to handle small details in dimensions of one or two millimetres, as well as dimensions of several metres. This wide range of the geometry necessitates a large number of cells in the numerical scheme, which increases the prediction cost and computing time to a rather high level. The problem is overcome by applying different simplifications such as *simplified boundary conditions*, *box method*, *prescribed velocity method* or *momentum method* see (Nielsen 1992). Continuous development of computational capacity and speed will undoubtedly make the direct methods with local grid refinements or multigrid solution possible. This is illustrated in Figure 1 with a diffuser consisting of 12 small slots which can be adjusted to different flow directions.

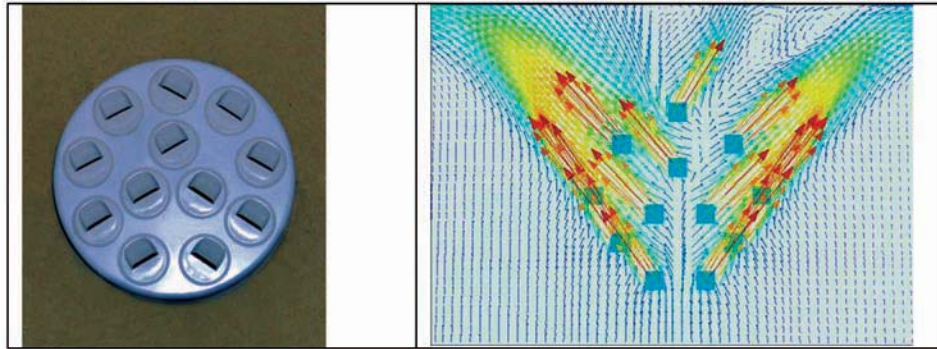


Figure 1- A direct simulation with a diffuser consisting of 12 small slots which can be adjusted to different flow directions. Development of computer capacity will make direct simulations of diffusers possible in the future.

The chapter does also discuss other boundary conditions such as surface boundary which is important for heat transfer predictions, free boundary, plane of symmetry, air exit opening and obstacle boundary.

The chapter on *Quality Control* is one of the more important chapters in the guidebook. Quality control consists of four major steps: to recognize possible error sources, to check for them in the simulation, to estimate the accuracy of the simulations, and to improve the simulation whenever possible. Two of the many examples given in the guidebook will be shown here.

It is a strong reduction in computing cost to work with two-dimensional flow instead of three-dimensional flow, but is it possible in all situations where the boundary conditions are “two-dimensional”? Figure 2 shows a long hall with a shed roof. There is complain about strong downdraught, but a two-dimensional prediction is not able to show this effect. It is necessary to use a three-dimensional transient approach to predict the downdraught.

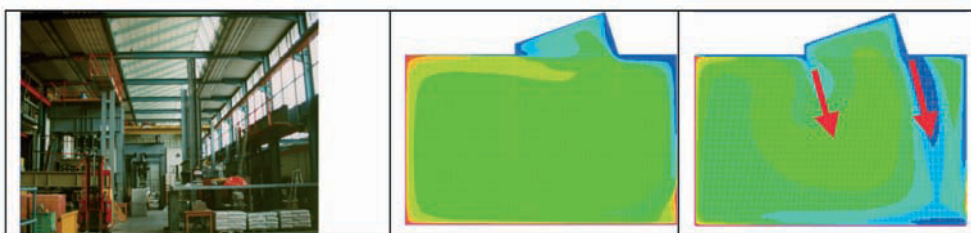


Figure 2 - Hall with shed roof, two-dimensional prediction and transient three-dimensional prediction.

For each set of problems a grid independence study must be performed. Figure 3 shows an example of such a study, where the same case (a transient fire simulation) is run for various grids from coarse to fine, and with homogeneous grids and other grids with mixtures of prisms, tetra- and hexahedral cells. The latter performs best. The figure shows the temperature distribution for 50,000 cells and 200,000 cells.

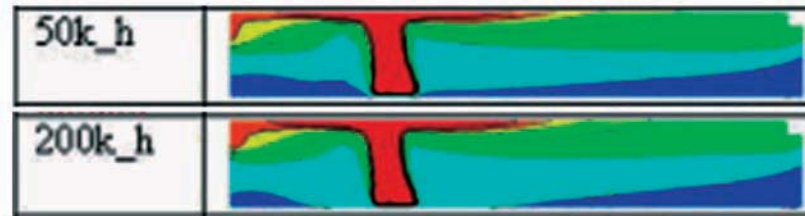


Figure 3 - Simulation of a transient fire with 50,000 and 200,000 cells in the solution domain.

The use of a CFD program in connection with other programs is discussed in the chapter *CFD combined with other prediction models*. Of highest interest and importance to ventilation design are the following: coupling of air flow and multi-zone dynamic thermal simulation, where especially energy storage is an important issue; coupling of air flow, moisture and energy transport through walls; coupling of air flow and multi-zone flow simulation, where the zonal flow simulation also handles the transport of additional components such as contaminants (e.g. smoke, CO₂, odours, moisture); and lastly coupling of the air flow and emission from building materials.

The possibilities for applying CFD for simulating the air flow in a building are discussed in the chapter *Application of CFD codes in building design*. A number of areas such as: room air movement, concentration distribution, emission from materials, thermal comfort assessment, ventilation effectiveness prediction and smoke management can be evaluated by a CFD program from the conceptual design to the preliminary design and right through to the final detail design stages.

2. CASE STUDIES


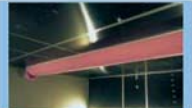


The chapter, *Case studies*, shows different practical application of predictions made at different stages in the initial design, detail design and commissioning phases. In particular, four different air distribution systems are studied by CFD and compared by measurements. The four systems are mixing ventilation with a wall-mounted diffuser, vertical ventilation, displacement ventilation and mixing ventilation generated by a ceiling mounted radial diffuser. All systems are designed to handle the same load in the same room, see Figure 4.



Figure 4 - Furnishings and heat load in the full-scale room. The heat load consists of two PCs, two desk lamps and two manikins producing a total heat load of 480 W.

The following Table 1 shows the different air distribution systems which will be studied.

Table 1. Four different air distribution systems.

Mixing ventilation with end wall mounted diffuser. Return opening below the supply.	
Vertical ventilation with a textile terminal. End wall mounted return opening at floor level.	
Displacement ventilation. End wall mounted low velocity diffuser. End wall mounted return opening below ceiling.	
Mixing ventilation generated by a ceiling mounted radial diffuser. End wall mounted return opening below ceiling.	

2.1. Mixing ventilation with end wall mounted diffuser

The inlet diffuser consists of 84 nozzles directing the air towards the ceiling in an angle of 40° upward. The diffuser is identical to the diffuser used in the IEA Annex 20 programme and is thus well known with regards both experiments and CFD (Chen and Moser 1991; Heikkinen 1991 and Fontaine et al. 1994).

Isothermal CFD simulations with two different geometric simplifications of the inlet diffuser have been performed (see Topp et al. (2001)). In both simplifications, air is supplied through one opening in the wall, and the geometry of the diffuser inside the room has thus been neglected. The inlet area in both models corresponds to that of the actual diffuser which means that the prescribed momentum flow is identical to the momentum flow in the real diffuser (*simplified boundary conditions*). The models investigated are: (A) Same width as actual diffuser, (B) A wider diffuser (1.5 x the width of actual diffuser).

The simulations show that both diffusers are able to predict the maximum velocity in the jet below the ceiling. None of the diffusers predict the width of the actual jet, although the predictions for diffuser B are better than those for diffuser A. Figure 5 shows the predictions for diffuser 3. It is obvious that it is necessary to have measurement data of the wall jet outlet from the diffuser, and it is also necessary to make several predictions to find the right boundary conditions.

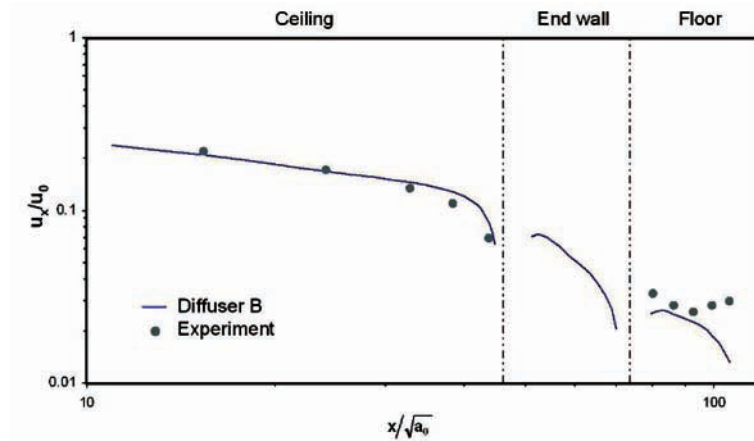


Figure 5 - Velocity decay in the recirculating isothermal flow for diffuser B (line) compared to experimental results (•). u_x is the maximum velocity at position x , u_0 is the inlet velocity, x is the position and a_0 is the effective inlet area. Both axes are in logarithmic scale.

2.2. Vertical Ventilation

The office room in Figure 4 is also ventilated by a vertical ventilation generated by a textile terminal device.

Table 1 shows the textile terminal which is designed as half a cylinder ($d = 315$ mm) located close to the ceiling. It is not possible to use cylindrical boundary conditions in the CFD program applied, so the first step in the design process is to generate an assumption for the inlet boundary conditions in Cartesian coordinates, and to make a calibration of the boundary conditions based on comparisons with measurements.

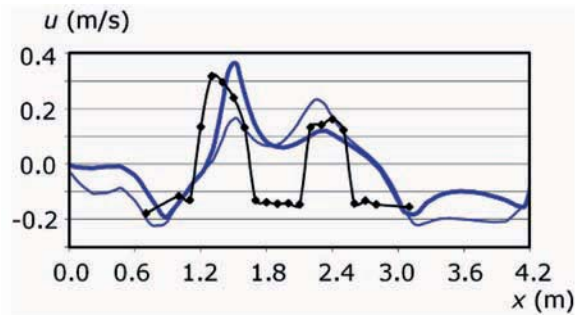


Figure 6 - Measured vertical velocity (♦) in the centre line of the room at the height of 1.8 m. The room is equipped with one desk in the symmetry plane. The blue lines are the predicted velocities in the symmetry plane, and 10 cm from the symmetry plane (thick line). x is the horizontal distance.

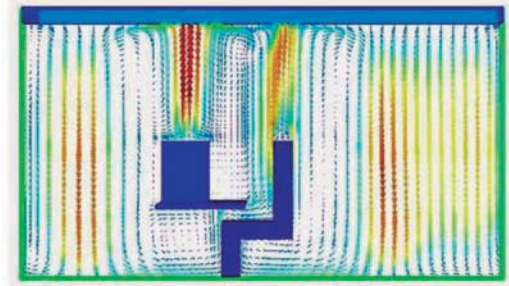


Figure 7 - Velocity distribution in the centre plane of an office room ventilated by a ceiling mounted textile terminal.

Figure 6 shows the predicted velocities in the centre plane of the room at the height of 1.8 m. The room is equipped with one desk in the symmetry plane. The predictions of the plumes above the PC ($x = 1.4$ m) and above the person ($x = 2.3$ m) are in good agreement with the measurements, but it is not possible to predict the downward-directed flow between the PC and the person which is shown in the experiments. Figure 7 shows the total velocity distribution in the middle plane. The results are in general in good agreement with measurements made by smoke experiments.

2.3. Displacement Ventilation

It is necessary to find a set of *simplified boundary conditions* which give the same flow along the floor as obtained by the actual diffuser in Table 1. The obstacles and the heat load are in this case located in the symmetry plane of the room. The diffuser is also located in the symmetry plane. The stratified flow from a displacement ventilation diffuser often moves in a symmetrical pattern along the floor, and therefore it is possible to solve the flow in only half of the room and save grid points in the predictions. This is only possible in the displacement ventilation case. Mixing ventilation and vertical ventilation may give unsymmetrical flow in rooms with symmetrical boundary conditions.

The stratified flow along the floor and the temperature distribution are shown in Figure 8. The results are very typical of displacement ventilation. The gravity effect on the cold air from the diffuser and the hot plumes from the heat sources are obvious in the figure. Also it is easy to see the constantly rising temperature with the height in the room.

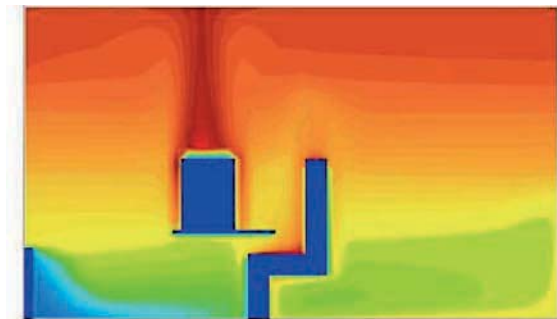


Figure 8 - Temperature distribution in the symmetry plane of a room with displacement ventilation.

2.4. Ceiling Mounted Radial Diffuser

In this case it is also necessary to do some initial work to establish a correct set of boundary conditions for the radial diffuser given in Table 1. Measurements show that the flow is radial with constant velocity in all directions. The best way to generate the radial flow in the CFD code is to build a diffuser with a horizontal surface which deflects the flow, much like the design in the real diffuser shown in Table 1. See Figure 9.

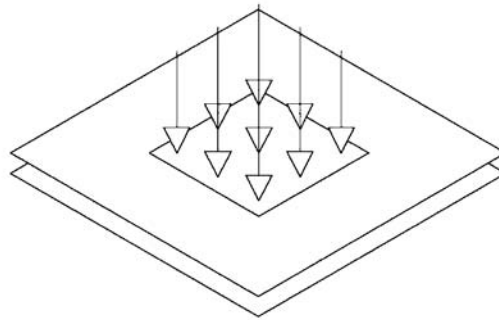


Figure 9 - The simulation of a radial diffuser. The vertical flow is deflected by a horizontal rectangular surface.

Figure 10 shows the measured and predicted velocity in a vertical section of the room close to the side walls at a height of 1.8 m. Measurements and predictions apply to the case of an air change rate of $n = 5.1 \text{ h}^{-1}$ and a temperature difference between return and supply of $\Delta T = 6.7\text{K}$.

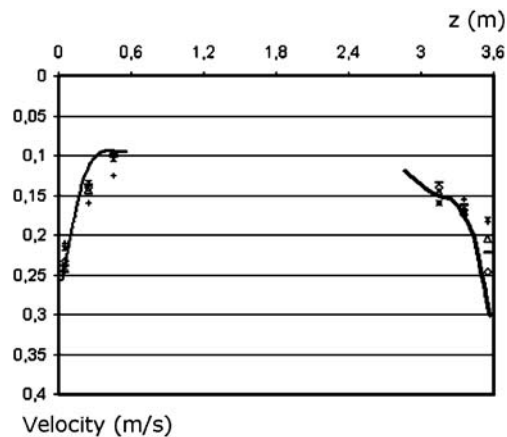


Figure 10 - Measured (points) and predicted (line) velocity distribution in the wall jet close to the side walls of the room.

In some situation the flow pattern may not only be turbulent but also slightly unsteady (time dependent). The work with CFD predictions shows that it is very difficult to obtain converged solutions in those cases with a set of steady equations, while the

solution of the time dependent (transient) equations is able to converge. Figure 11 shows the time dependent solution of the velocity at a height of 1.8 m for four different time steps in a case where the measurements also show an unsteady flow.

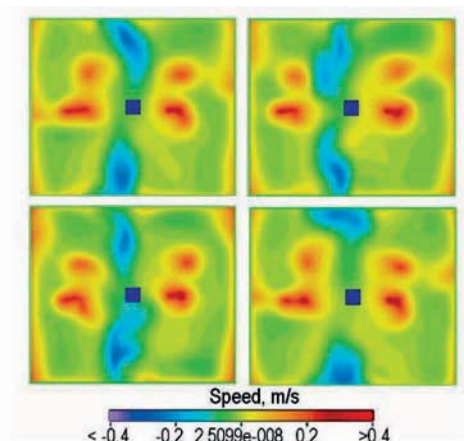


Figure 11 - Vertical velocities at the height of 1.8 m at four different time steps indicated in clockwise direction, $\Delta t = 4$ s.

The experiments with the different air distribution systems are reported by (Jacobsen et al. 2002), (Nielsen et al. 2005) and (Nielsen et al. 2006).

The final chapter in the guidebook discusses different types of benchmark tests. A benchmark test can be used for beginners within CFD to obtain a quick insight into different problems with the prediction of ventilation, and to obtain an initial experience by comparing CFD output.

See www.cfd-benchmarks.com

3. CONCLUSION

The REHVA Guidebook aims to provide engineers with a comprehensive and easy-to-understand publication on Computational Fluid Dynamics.

The guidebook is written for people who have to use and evaluate predictions based on a CFD program and it gives a quick insight into the subject for people who are not used to work with CFD. A large number of CFD predictions are made nowadays and it is often difficult to judge the quality level of the predictions. The guidebook introduces rules for good prediction work, explains the principles behind the numerical method, and discusses the special subjects behind ventilation. It is the hope that it will improve the technical level of the CFD work.

The guidebook shows the CFD predictions of a number of practical cases, which are compared with experiments. It is obvious that a CFD prediction is an efficient tool for design of room air distribution.

REFERENCES

- Chen, Q. and Moser, A. 1991. Simulation of a multiple-nozzle diffuser. *Proceedings of the 12th AIVC Conference on Air Movement and Ventilation Control within Buildings*, Ottawa, Canada, Vol. 2, 1-13.
- Fontaine, J.R., Biolley, F., Rapp, R., Sérieys, J. C. and Cunin, J.C. 1994. Analysis of a three-dimensional ventilation flow: Experimental validation on a water scale model of numerical simulations. *Numerical Heat Transfer, Part A*, Vol. 26, 431-451.
- Heikkinen, J. 1991. Modelling of a supply air terminal for room air flow simulation. *Proceedings of the 12th AIVC Conference on Air Movement and Ventilation Control within Buildings*, Ottawa, Canada, Vol. 3, 213-230.
- Jacobsen, T.S., Hansen, R., Mathiesen, E., Nielsen, P.V. and Topp, C. 2002. Design methods and evaluation of thermal comfort for mixing and displacement ventilation. 8th International Conference on Air Distribution in Rooms, ROOMVENT 2002, Copenhagen.
- Nielsen, P.V., Heby, T. and Moeller-Jensen, B. 2006. Air distribution in a room with ceiling-mounted diffusers - comparison with wall-mounted diffuser, Vertical Ventilation and Displacement Ventilation. *ASHRAE Transactions 2006*, Vol. 112, Part 2.
- Nielsen, P.V., Topp, C., Soennichsen, M. and Andersen, H. 2005. Air distribution in rooms generated by a textile terminal - comparison with mixing ventilation and displacement ventilation. *ASHRAE Transactions*, Vol. 111, Part 1, pp. 733-739.
- Nielsen, P.V. (ed.), Francis Allard, Hazim B. Awbi, Lars Davidson, Alois Schäljinj. 2007. Computational fluid dynamics in ventilation design. *REHVA Guidebook No. 10*, ISBN 2-9600468-9-7, Finland.
- Nielsen, P.V. 1992. The description of supply openings in numerical models for room air distribution. *ASHRAE Transactions*, Vol. 98, Part 1, 1992.
- Topp, C., Jensen, R.L., Pedersen, D.N. and Nielsen, P.V. 2001. Validation of boundary conditions for CFD simulations on ventilated rooms. *Proceedings of the 4th International Conference on Indoor Air Quality, Ventilation & Energy Conservation*, Vol. 1, pp. 295-302, Hunan University.